Combination of boundary and finite elements in elastostatics

C. A. Brebbia and P. Georgiou

Department of Civil Engineering, University of Southampton, Southampton SO9 5NH, England
(Received April 1978)

Introduction

Renewed interest has recently been focused on the application of boundary solutions in preference to domain type solutions such as finite elements for the analysis of two- and three-dimensional problems. These techniques are known under different names, such as integral equations, boundary integral methods, etc. and Rizzo[1] seems to have been the first to have used them in classical elastostatics, the work being continued by Cruse[2,3] who extended it to elastodynamics both of them basing their work on the so-called direct integral formulation, as opposed to the indirect formulation which was used by Tomlin[4] to solve the problem of zoned anisotropic media. All these papers are based on using constant values for the displacements and tractions over part of the boundary.

Lachat,[5] following the work of the Southampton University group, developed a technique which allowed for variations of displacements and tractions along parts of the boundary surface or 'elements' and hence the term 'boundary elements' was adopted to define this particular method.[6] The idea of using interpolation functions to define the variables along these elements is important as it allows for the combination of finite and boundary element regions without any loss of continuity. In addition the work at Southampton[6] concentrated on the common basis of the different methods, how they could be interpreted as special cases of the weighted residual formulation,[7] and the equivalence of the direct and indirect boundary element techniques.[8]

The idea of combining both techniques can be attributed to Wexler,[9] who started to use integral equation solutions to represent the unbounded field problem early in the 1970s, the advantage being that this allowed for the use of appropriate conditions to represent the infinite domain. The integral equation technique is also of interest when regions of high stress or potential gradients exist, but finite elements are adequate for other parts of a body and may be simpler to use in cases such as layered continuum, anisotropic and nonlinear materials. Hence it is important for the analyst to be able to represent a body using finite or boundary element techniques, depending on the particular geometry, boundary conditions, etc.

The first combination of the two methods for elastostatics appears to be by Osias,[10] although for wave propagation problems, the method was used by Mei[11] in 1975 who explained the way of combining both solutions using variational techniques.

This paper examines the combination of boundary and finite elements for two-dimensional elastostatic problems, using two different approaches. The first method treats the boundary element region as a finite element and the second treats the finite element region as an equivalent boundary element region. The first appears to be more interesting as it can easily be incorporated into existing finite element codes.

The technique is applied to a series of examples to illustrate how the combination can be done and how accurate the results are.

Basic relations

The principle of virtual displacements for linearly elastic materials can be written as:

$$\int_{\Omega}(\sigma_{jk,j} + b_k)u_k^* \,\mathrm{d}\Omega = \int_{\Gamma_2}(p_k - \bar{p}_k)u_k^* \,\mathrm{d}\Gamma \tag{1}$$

where:

$$(\ ),j = \frac{\partial(\ )}{\partial xj}$$

and u_k^* are the virtual displacements satisfying the homogeneous boundary conditions, $\bar{u}_k^* = 0$, on Γ_1.

σ_{jk} are the components of the stress tensor, b_k, the body forces and p_k are the tractions on the boundary Γ, given by $p_k = n_j\sigma_{jk}$, where n_j are the direction cosines. The bar indicates known quantities.

If we now interpret u_k^* as a weighting function which does not identically satisfy the boundary conditions on Γ_1, equation (1) becomes:

$$\int_{\Omega}(\sigma_{jk,j} + b_k)u_k^* \,\mathrm{d}\Omega = \int_{\Gamma_2}(p_k - \bar{p}_k)u_k^* \,\mathrm{d}\Gamma + \int_{\Gamma_1}(\bar{u}_k - u_k)p_k^* \,\mathrm{d}\Gamma \tag{2}$$

0307-904X/79/030212-09 $02.00

Note that the surface tractions corresponding to the u_k^* system of displacements are:

$$p_k^* = n_j \sigma_{jk}^* \tag{3}$$

We now assume linear strain-displacement relations, i.e.:

$$\epsilon_{ij} = \frac{1}{2}\left\{\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right\} = \tfrac{1}{2}\{u_{i,j} + u_{j,i}\} \tag{4}$$

$$\epsilon_{ij}^* = \frac{1}{2}\left\{\frac{\partial u_i^*}{\partial x_j} + \frac{\partial u_j^*}{\partial x_i}\right\} = \tfrac{1}{2}\{u_{i,j}^* + u_{j,i}^*\} \tag{5}$$

and that the material properties are also linear, which gives:

$$\int_\Omega \sigma_{ij}\epsilon_{ij}^* \, d\Omega = \int_\Omega \sigma_{ij}^* \epsilon_{ij} \, d\Omega \tag{6}$$

Hence we can now integrate (2) by parts which gives:

$$\int_\Omega b_k u_k^* \, d\Omega - \int_\Omega \sigma_{jk}\epsilon_{jk}^* \, d\Omega$$

$$= -\int_{\Gamma_2} \bar{p}_k u_k^* \, d\Gamma - \int_{\Gamma_1} p_k u_k^* \, d\Gamma + \int_{\Gamma_1} (\bar{u}_k - u_k) p_k^* \, d\Gamma \tag{7}$$

Integrating by parts once more, one obtains:

$$\int_\Omega b_k u_k^* \, d\Omega + \int_\Omega \sigma_{jk,j}^* u_k \, d\Omega$$

$$= -\int_{\Gamma_2} \bar{p}_k u_k^* \, d\Gamma - \int_{\Gamma_1} p_k u_k^* \, d\Gamma + \int_{\Gamma_1} \bar{u}_k p_k^* \, d\Gamma + \int_{\Gamma_2} u_k p_k^* \, d\Gamma \tag{8}$$

The starting integral relationship (2), which is an integral in the domain Ω can now be reduced to an integral on the boundary Γ (i.e. passing from a domain to a boundary problem), by finding an analytical solution which makes the second integral in equation (8) equal to zero. The most convenient one is the fundamental solution.

Note that expression (7) is the usual starting expression for the finite element technique, although in the finite element method, the integrals on Γ disappear as the boundary conditions $\bar{u}_k = u_k$ are easy to satisfy identically.

Fundamental solution

By using the fundamental solution to eliminate the domain integral term in (8) the problem is reduced to a boundary integral and can be solved numerically by the discretization of the boundary into elements. This gives rise to the boundary element method.

The fundamental solution is the one that satisfies the equation:

$$\sigma_{jk,j}^* + \Delta_l^i = 0 \tag{9}$$

where Δ_l^i is the Dirac delta function and represents a unit load at the internal point 'i' in the 'l' direction. This type of solution will produce for each direction 'l' the following equation:

$$u_l^i + \int_{\Gamma_1} \bar{u}_k p_k^* \, d\Gamma + \int_{\Gamma_2} u_k p_k^* \, d\Gamma$$

$$= \int_\Omega b_k u_k^* \, d\Omega + \int_{\Gamma_1} p_k u_k^* \, d\Gamma + \int_{\Gamma_2} \bar{p}_k u_k^* \, d\Gamma \tag{10}$$

where u_l^i represents the displacement at 'i' in the 'l' direction. In general we can write for point 'i':

$$u_l^i + \int_\Gamma u_k p_k^* \, d\Gamma = \int_\Gamma p_k u_k^* \, d\Gamma + \int_\Omega b_k u_k^* \, d\Omega \tag{11}$$

where:

$$\Gamma = \Gamma_1 + \Gamma_2$$

Note that u_k^* and p_k^* are the fundamental solutions, i.e. the displacements and tractions due to a concentrated unit load at point 'i' in the direction 'l'. If we consider unit forces acting in the three directions, (10) can be written as:

$$u_l^i + \int_\Gamma u_k p_{lk}^* \, d\Gamma = \int_\Gamma p_k u_{lk}^* \, d\Gamma + \int_\Omega b_k u_{lk}^* \, d\Omega \tag{12}$$

where p_{lk}^* and u_{lk}^* represent the tractions and displacements in the 'k' direction due to a unit force in the 'l' direction. Equation (11) is valid for the particular point 'i' where these forces are applied.

The fundamental solution for a two-dimensional isotropic body in plane strain is:

$$u_{lk}^* = \frac{1}{8\pi G(1-\nu)}\left[(3-4\nu)\ln\left(\frac{1}{r}\right)\Delta_{lk} + \frac{\partial r}{\partial x_l}\cdot\frac{\partial r}{\partial x_k}\right]$$

$$p_{lk}^* = \frac{1}{4\pi(1-\nu)r}\left[\frac{\partial r}{\partial n}\cdot\left\{(1-2\nu)\Delta_{kl} + 2\frac{\partial r}{\partial x_k}\frac{\partial r}{\partial x_l}\right\} - (1-2\nu)\left\{\frac{\partial r}{\partial x_l}\cdot n_k - \frac{\partial r}{\partial x_k}\cdot n_l\right\}\right] \tag{13}$$

where: p_{lk}^* and u_{lk}^* are defined as above; G is the shear modulus of the material; ν, the poisson ratio; r, distance between observation point and point of application of the unit load and Δ_{lk}, Dirac delta function, represents a unit load in the 'l' direction applied at point 'k'.

Matrix formulation

In the particular case we are considering, i.e. that of a two-dimensional body undergoing plane strain or plane stress, k and l in the above expressions take the values 1 to 2. The formulations above may be expressed in a matrix notation as opposed to the summation convention used.

In matrix form $\boldsymbol{u}^*$ is a 2×2 matrix with elements u_{lk}^* ($l, k = 1, 2$), as is $\boldsymbol{p}^*$, with elements p_{lk}^*, i.e.:

$$\boldsymbol{p}^* = \begin{bmatrix} p_{11}^* & p_{12}^* \\ p_{21}^* & p_{22}^* \end{bmatrix} \quad \boldsymbol{u}^* = \begin{bmatrix} u_{11}^* & u_{12}^* \\ u_{21}^* & u_{22}^* \end{bmatrix} \tag{14}$$

The unknown displacements and tractions, and the known body forces may be written as vectors:

$$\boldsymbol{u} = \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} \quad \boldsymbol{p} = \begin{Bmatrix} p_1 \\ p_2 \end{Bmatrix} \quad \boldsymbol{b} = \begin{Bmatrix} b_1 \\ b_2 \end{Bmatrix} \tag{15}$$

The basic boundary equation (12) can be expressed in matrix form as:

$$c^i u^i + \int_\Gamma \boldsymbol{p}^* \boldsymbol{u} \, d\Gamma = \int_\Gamma \boldsymbol{u}^* \boldsymbol{p} \, d\Gamma + \int_\Omega \boldsymbol{u}^* \boldsymbol{b} \, d\Omega \tag{16}$$

The constants c^i are generally determined from the rigid body conditions.

Boundary elements

Consider the case of the boundary values of $\boldsymbol{u}$ and $\boldsymbol{p}$ given by some interpolation functions, such that:

$$\begin{aligned} \boldsymbol{u} &= \Phi^T \boldsymbol{u}^n \\ \boldsymbol{p} &= \Psi^T \boldsymbol{p}^n \end{aligned} \tag{17}$$

The simplest possible elements are those for which $\boldsymbol{u}$ and $\boldsymbol{p}$ are constant over the element and consist of a straight line with a central node. The value of $\boldsymbol{p}$ and $\boldsymbol{u}$ over the whole element is taken as constant and equal in value to that at the node. In general $\boldsymbol{u}$ and $\boldsymbol{p}$ can have any variation simply by choosing the appropriate interpolation functions Φ and Ψ. These functions are standard interpolation functions similar to those used in finite element formulations, the main difference being that they vary only along the boundary Γ as opposed to over the domain Ω, which in fact lessens their complexity. In order to find the body force terms the domain has to be divided into a series of cells (*Figure 2*), or internal elements, however, in contrast to finite elements this process does not introduce any internal unknowns.

Substituting (17) into the matrix equation (16), we can write for each particular node, 'i':

$$c^i \boldsymbol{u}^i + \sum_{l=1}^{N} \left\{ \int_{\Gamma_l} \boldsymbol{p}^* \Phi^T \mathrm{d}\Gamma \right\} \boldsymbol{u}^n = \sum_{l=1}^{N} \left\{ \int_{\Gamma_l} \boldsymbol{u}^* \Psi^T \mathrm{d}\Gamma \right\} \boldsymbol{p}^n + \sum_{k=1}^{M} \left\{ \int_{\Omega_k} \boldsymbol{u}^* \boldsymbol{b} \, \mathrm{d}\Omega \right\} \tag{18}$$

where: N is the number of boundary elements; M is the number of internal cells; Γ_l is the surface of the 'l' boundary element and Ω_k is the area of the 'k' internal cell.

The interpolation functions are expressed in some local, homogeneous system of coordinates and the integrals are carried out numerically. For each node 'i' the integrals relate this node to the others, 'j', via the fundamental solution $\boldsymbol{p}^*$. ($\boldsymbol{p}^*$ is dependent on the distance between the two nodes – see equation (13).) In this case the integrals are 2×2 matrices, as there are two unknowns at each node, and equation (18), for each node 'i', may now be written in matrix notation as:

$$c^i \boldsymbol{u}^i + \sum_{j=1}^{NN} \hat{\boldsymbol{H}}_{ij} \boldsymbol{u}_j = \sum_{j=1}^{NN} \boldsymbol{G}_{ij} \boldsymbol{p}_j + \boldsymbol{b}_i \tag{19}$$

where NN is the number of nodes and $\hat{\boldsymbol{H}}_{ij}$ and $\boldsymbol{G}_{ij}$ are the $2 \times NN$ matrices produced by the integrations in equation (18). This equation relates the value at the node 'i' to the value at each of the other nodes, including 'i'.

One can write equation (19) for each of the 'i' nodes, obtaining in total $2 \times NN$ equations, corresponding to $2 \times NN$ unknowns. Let us now call:

$$\begin{aligned} \boldsymbol{H}_{ij} &= \hat{\boldsymbol{H}}_{ij} && \text{for } i \neq j \\ \boldsymbol{H}_{ij} &= \hat{\boldsymbol{H}}_{ij} + \boldsymbol{C}_i && \text{for } i = j \end{aligned} \tag{20}$$

where $\boldsymbol{C}^i$ is a coefficient matrix due to the boundary geometry, i.e.:

$$C^i = \begin{bmatrix} c & 0 \\ 0 & c \end{bmatrix} \tag{21}$$

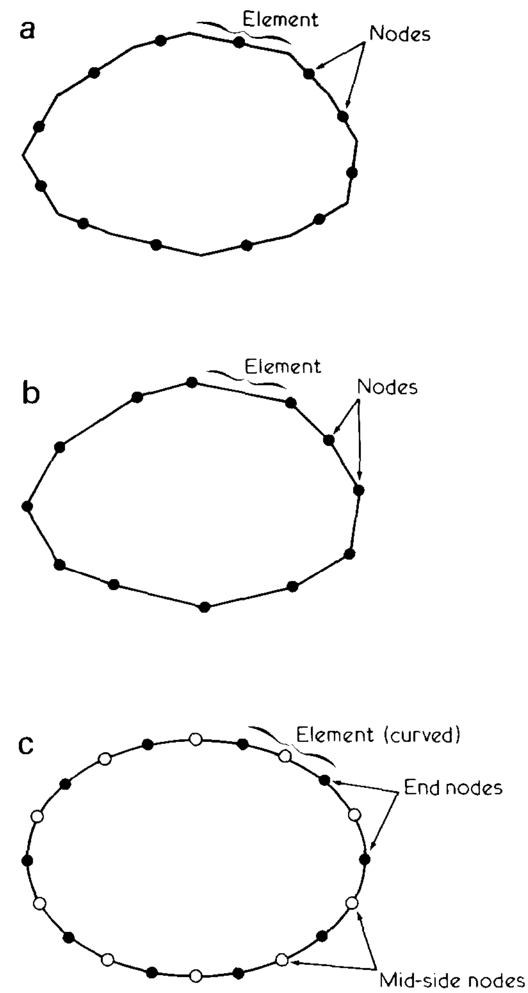

Figure 1 Two-dimensional body divided into boundary elements

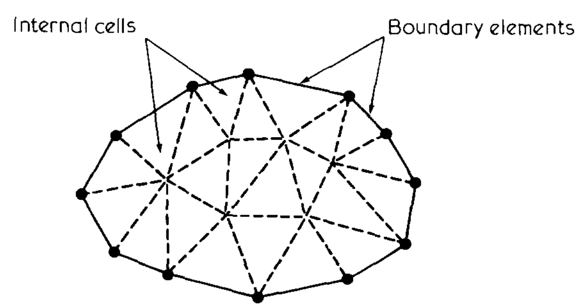

Figure 2 Body divided into boundary elements and internal cells

(Note that $C = \frac{1}{2}$ for smooth boundaries.)

Hence equation (19) can now be expressed as:

$$\sum_{j=1}^{NN} \boldsymbol{H}_{ij} \boldsymbol{u}_j = \sum_{j=1}^{NN} \boldsymbol{G}_{ij} \boldsymbol{p}_j + \boldsymbol{b}_i \tag{22}$$

The whole set of equations for the NN boundary nodes can be expressed in matrix form as:

$$\boldsymbol{HU} = \boldsymbol{GP} + \boldsymbol{B} \tag{23}$$

Note that $N1$ values of displacements and $N2$ values of tractions ($NN = N1 + N2$) are known on the boundary, and hence in the $\boldsymbol{U}$ and $\boldsymbol{P}$ vectors, there remain NN unknowns, which may all be gathered into a left-hand-side vector $\boldsymbol{X}$, after reordering the equations, to obtain:

$$\boldsymbol{AX} = \boldsymbol{F} + \boldsymbol{B} \tag{24}$$

Equation (24) may now be solved to yield all remaining unknown displacements and tractions on the boundary, upon which the displacements and stresses at any point within the interior of the domain may be calculated using the expressions:

$$u^i = \int_\Gamma u^* p \, d\Gamma - \int_\Gamma p^* u \, d\Gamma + \int_\Omega u^* b \, d\Omega \tag{25}$$

$$\sigma_{ij} = \int_\Gamma D_{ij} p \, d\Gamma - \int_\Gamma S_{ij} u \, d\Gamma + \int_\Omega D_{ij} b \, d\Omega \tag{26}$$

where:

$$\begin{aligned} D_{ij} &= [D_{1ij} D_{2ij}] \\ S_{ij} &= [S_{1ij} S_{2ij}] \end{aligned} \tag{27}$$

The values of the coefficients are:

$$D_{kij} = \frac{1}{r^2}\left\{(1-2\nu)\left[\Delta_{ki}\frac{\partial r}{\partial x_j} + \Delta_{kj}\frac{\partial r}{\partial x_i} - \Delta_{ij}\frac{\partial r}{\partial x_k}\right] + 2\frac{\partial r}{\partial x_i}\cdot\frac{\partial r}{\partial x_j}\cdot\frac{\partial r}{\partial x_k}\right\}\cdot\frac{1}{4\pi(1-\nu)} \tag{28}$$

$$\begin{aligned} S_{kij} = \frac{G}{r}\Bigg\{ & 2\cdot\frac{\partial r}{\partial n}\left[(1-2\nu)\Delta_{ij}\frac{\partial r}{\partial x_k} + \nu\left(\Delta_{ik}\frac{\partial r}{\partial x_j} + \Delta_{jk}\frac{\partial r}{\partial x_i}\right)\right. \\ & \left. - 4\frac{\partial r}{\partial x_i}\cdot\frac{\partial r}{\partial x_j}\cdot\frac{\partial r}{\partial x_k}\right] + 2\nu\left(n_i\frac{\partial r}{\partial x_j}\frac{\partial r}{\partial x_k} + n_j\frac{\partial r}{\partial x_i}\frac{\partial r}{\partial x_k}\right) \\ & + (1-2\nu)\left[2n_k\frac{\partial r}{\partial x_i}\frac{\partial r}{\partial x_j} + n_j\Delta_{ik} + n_i\Delta_{jk}\right] \\ & - (1-4\nu)n_k\Delta_{ij}\Bigg\}\frac{1}{4\pi(1-\nu)} \end{aligned} \tag{29}$$

Relationship between finite and boundary elements

In many cases it is convenient to relate finite elements to a boundary element region. The relationship between the two techniques becomes clear when the fundamental relations used to develop them are examined. In the above formulation equation (7) is also the usual starting point for the finite element technique, which assumes that the boundary conditions on Γ_1 are identically satisfied and that u_k^* is the virtual displacement field which can be expressed in terms of nodal values using the same interpolation functions as those used for the approximate, assumed field, u_k.

Under these conditions the standard finite element system may readily be obtained:

$$KU = F + D \tag{30}$$

where: K is the stiffness matrix for the system, F the equivalent nodal force vector, and D the vector due to body force.

The vector F is obtained by weighting the applied tractions $\bar{p}_k$ on Γ_2 by the interpolation functions used for the displacements, i.e.:

$$u^{*n,T}F = \Sigma\left\{\int_{\Gamma_2} \bar{p}_k u_k^* \, d\Gamma\right\} \tag{31}$$

where $u^{*n,T}$ is the vector of nodal virtual displacements for the complete system. The summation applies over the element sides on the boundary. Under these conditions we can write F as:

$$F = MP \tag{32}$$

where P is a vector of nodal tractions and where M is a matrix due to the weighting of the boundary tractions by the interpolation functions for the displacements. The exact nature of M can be simply explained by considering the term:

$$\int_{\Gamma_2} \bar{p}_k u_k^* \, d\Gamma \tag{33}$$

in equation (7), which is the boundary tractions weighted by the virtual displacements, or may be interpreted as the external energy expended due to a virtual displacement u^*. For a particular element, 'l', P_k^T is a row vector containing the nodal value of the tractions and u_k^* a column vector containing the arbitrary virtual displacements u^*. Equation (33) may now be written as:

$$\int_{\Gamma_l} u_k^{*,T} p_k \, d\Gamma \tag{34}$$

Assuming the interpolation functions for u and p as:

$$\begin{aligned} u_k^* &= \Phi u_k^{*,n} \\ p_k &= \Psi p_k^n \end{aligned} \tag{35}$$

The above term reduces to:

$$u^{*n,T}\left\{\int_{\Gamma_l} \Phi\Psi^T d\Gamma\right\}\cdot p_k^n \tag{36}$$

The matrix M may then be found from:

$$U^*MP = \Sigma\left[u^{*n,T}\left\{\int_{\Gamma_l} \Phi\Psi^T\cdot d\Gamma\right\} p_k^n\right] \tag{37}$$

Hence we can express (30) as:

$$KU = MP + D \tag{38}$$

which is a similar form to the boundary element equation (23).

Consider a problem consisting of two domains Ω^1, Ω^2 joined by an interface Γ^I, and which makes use of a finite element formulation in Ω^2 and a boundary element formulation in Ω^1 (*Figure 3*). In order to join the two parts we apply compatibility and equilibrium conditions along the interface Γ_I, i.e.:

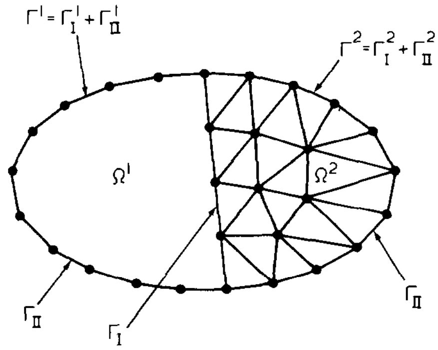

Figure 3 Body divided into finite elements and boundary elements

$$U_I^1 = U_I^2 \tag{39}$$

$$P_I^1 + P_I^2 = 0 \tag{40}$$

where u_I^l, p_I^l refer to the displacements and tractions on the interface Γ_I for the region l ($l = 1, 2$).

We now have two alternatives as to how to approach the problem. We may develop the boundary element region Ω^1 as an equivalent finite element, assemble the effective stiffness matrix with those of the finite elements of region Ω^2 and solve the overall system as a stiffness problem.[13] Alternatively we can consider Ω^1 and Ω^2 as if they were both boundary element formulations. This is the equivalent boundary element approach.

Using the first approach, we can transform equation (23) by inverting G, such that:

$$G^{-1}(HU - B) = P \tag{41}$$

and premultiply by the matrix M described in (32), giving:

$$(MG^{-1}H)U - (MG^{-1}B) = MP \tag{42}$$

We can now define:

$$\begin{aligned} K' &= MG^{-1}H \\ D' &= MG^{-1}B \\ F' &= MP \end{aligned} \tag{43}$$

Hence equation (42) has the following finite element form:

$$K'U = F' + D' \tag{44}$$

The main discrepancy which arises with the above formulation is the fact that the matrix K' is generally asymmetric although from first principles a stiffness matrix should be symmetric. The asymmetry arises due to the approximations involved in the discretization process and the choice of the assumed solution. The matrix can be made symmetric by minimizing the square of the errors in the non-symmetric off-diagonal terms.

The error for a coefficient ij can be written as the difference between k'_{ij} and k'_{ji} and the still unknown coefficient k_{ij}, which is symmetric, i.e.:

$$\epsilon_{ij} = \tfrac{1}{2}\{(k_{ij} - k'_{ij}) + (k_{ij} - k'_{ji})\} \tag{45}$$

The square of this error is now minimized:

$$\frac{\partial}{\partial k_{ij}}(\epsilon_{ij}^2) = 2k_{ij} - k'_{ji} - k'_{ij} = 0 \tag{46}$$

Hence the new symmetric coefficients are:

$$k_{ij} = \tfrac{1}{2}(k'_{ij} + k'_{ji}) \tag{47}$$

The equivalent finite element type matrices of equation (44) may now be assembled with the matrices for the elements of region 2 to form a global system of equations. Assembling along the nodes of interface I ensures condition (39) and as the matrix M transforms the tractions P into equivalent consistent nodal loads F, by having zeros in the vector F corresponding to the nodes u_I, condition (40) is satisfied.

Using the second approach, mentioned above, for combining the two methods, we can consider region 2 as a boundary element type region.

For region 1 we can write:

$$[H^1 \;\; H_I^1]\begin{Bmatrix} U^1 \\ U_I^1 \end{Bmatrix} = [G^1 \;\; G_I^1]\begin{Bmatrix} P^1 \\ P_I^1 \end{Bmatrix} + B^1 \tag{48}$$

and for region 2:

$$[K^2 \;\; K_I^2]\begin{Bmatrix} U^2 \\ U_I^2 \end{Bmatrix} = [M^2 \;\; M_I^2]\begin{Bmatrix} P^2 \\ P_I^2 \end{Bmatrix} + D^2 \tag{49}$$

By writing $P_I = P_I^1 = -P_I^2$ and $U_I = U_I^1 = U_I^2$ we automatically satisfy conditions (39) and (40), and equations (48) and (49) can be rearranged as follows:

$$[H^1 \;\; H_I^1 \;\; -G_I^1]\begin{Bmatrix} U^1 \\ U_I \\ P_I \end{Bmatrix} = [G^1]\{P^1\} + B^1 \tag{50}$$

and

$$[K^2 \;\; K_I^2 \;\; M_I^2]\begin{Bmatrix} U^2 \\ U_I \\ P_I \end{Bmatrix} = [M^2] \; P^2 + D^2 \tag{51}$$

Writing these two equations together, as a single matrix equation, we have:

$$\begin{bmatrix} H^1 & H_I^1 & -G_I^1 & 0 \\ 0 & K_I^2 & M_I^2 & K^2 \end{bmatrix}\begin{Bmatrix} U^1 \\ U_I \\ P_I \\ U^2 \end{Bmatrix} = \begin{bmatrix} G^1 & 0 \\ 0 & M^2 \end{bmatrix}\begin{Bmatrix} P^1 \\ P^2 \end{Bmatrix} + \begin{Bmatrix} B^1 \\ D^1 \end{Bmatrix} \tag{52}$$

Notice that on the boundary of the finite element region Ω^2, only the displacements on Γ_1 have to be prescribed, whilst on the boundary of Ω^2 we prescribe the displacements or tractions and consequently need to re-order the equations.

The advantage of the second approach is that it does not require an inversion.

Applications

A few computer cases were run in order to examine the combined finite and boundary element solution technique in two-dimensional elastostatics. The programs developed combine constant boundary elements with quadratic finite elements and although this combination is not fully compatible, it gives good results in practice and also has the advantage of avoiding the corner problems which appear in boundary element solutions.[14]

Two basic problems were considered. The first consisted of two closed domains and was used to test the validity of the two types of combination method; and the second was considered in order to examine the use of a boundary element domain to represent a semi-infinite space.

The test problems considered were originally solved using the equivalent finite element approach described above and the effective stiffness matrix K' (given by equation (44)) for the boundary element region was examined before symmetrization was carried out. Although the simplest variation of U and P around the boundary was assumed in order to calculate the M matrix (i.e. u and p constant along each element), K' was found to be basically symmetric, most of the asymmetry occurring in some of the off-diagonal terms which were of much lower order than the governing diagonal coefficients. The matrix was then symmetrized and the similarity of the numerical results confirms the applicability of the least square symmetrization technique.

Example 1

A test problem used for the combination of the finite and boundary element techniques is shown in *Figure 4a*; the domain has been divided into two regions, the first of which is descretized into normal finite elements, and the second into boundary elements. The $\boldsymbol{H}$ and $\boldsymbol{G}$ matrices for the boundary region are used to form an equivalent stiffness matrix, as shown in equations (43), and this matrix is then combined with the matrix for the finite element region to form the global stiffness matrix.

Region 2 was considered separately as a check on the validity of the idea of formulating an equivalent stiffness matrix, as shown in equation (44), and this matrix is then is shown in *Figure 4b*.

This region was studied using the boundary element method, i.e. the $\boldsymbol{H}$ and $\boldsymbol{G}$ matrices were formed for the system, the equations re-ordered and solved to yield the unknown displacements and tractions. (Method 1, *Table 1*.)

Secondly, the matrix $\boldsymbol{K} = \frac{1}{2}(\boldsymbol{K}' + \boldsymbol{K}', \boldsymbol{T})$ was formed, where $\boldsymbol{K}' = (\boldsymbol{M} \cdot \boldsymbol{G}^{-1} \cdot \boldsymbol{H})$. The stiffness system $\boldsymbol{K} \cdot \boldsymbol{U} = \boldsymbol{P}$ was then solved to yield the unknown displacements. (Method 2, *Table 1*.)

The results obtained are tabulated below (*Table 1*) and clearly demonstrate the validity of using the least squares technique to symmetrize the stiffness matrix.

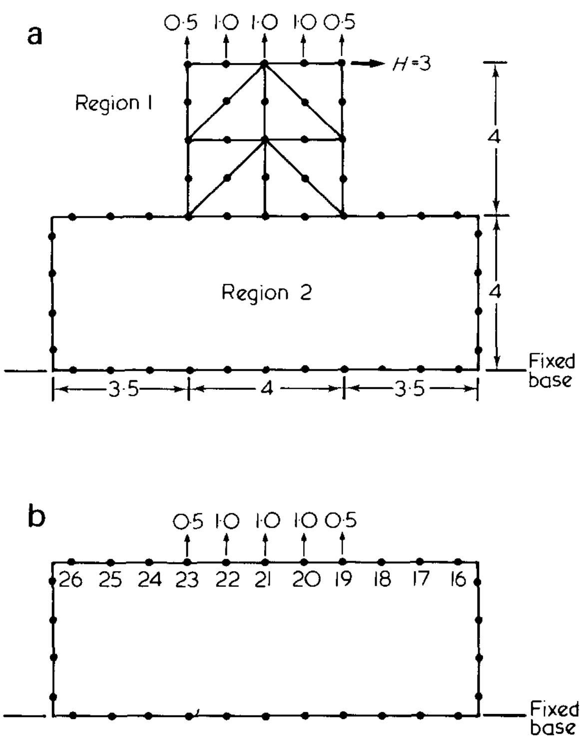

Figure 4 (a), Grid used for combination of finite and boundary element regions. (b), boundary element region

Table 1 $E = 2 \times 10^5$; $\nu = 0.2$

	Displacement $\times 10^{-6}$	
Node	Method 1. B.E.	Method 2. $\boldsymbol{KU} = \boldsymbol{P}$
16	0.070	0.071
17	0.224	0.224
18	0.549	0.549
19	1.304	1.304
20	1.605	1.605
21	1.684	1.684

The equivalent stiffness matrix for region 2 was then assembled with the finite elements of region 1 (*Figure 4a*). to form the overall system $\boldsymbol{KU} = \boldsymbol{P}$ the results are shown as Approach 1 in *Table 2*. A program to implement the second of the two alternative combination procedures (i.e. the equivalent boundary element approach – equation (52)) was then developed. The matrix $\boldsymbol{M}$ is formed by integration around the perimeter of the finite element region and the $\boldsymbol{M}, \boldsymbol{K}, \boldsymbol{H}, \boldsymbol{G}$ matrices are broken down and reassembled in an overall system as given by equation (52). The equations are then reordered and solved, the results are shown as Approach 2 in *Table 2*.

Two loading cases were considered, the first being a uniformly distributed vertical load equal to 4.0 on the top edge, producing a symmetrical problem, and the second included a concentrated horizontal load equal to 3.0 on the top corner node (see *Figure 4a*).

In order to make a comparative study, the system was also analysed as a whole, using both the finite element and boundary element techniques. The meshes used are shown in *Figures 5* and *6*, and the results obtained for displacements (which are all very similar) are tabulated in *Table 2*.

This application shows the validity of developing an equivalent stiffness matrix from a boundary element formulation. This matrix can also be symmetrized to make it compatible with existing finite element programs without loss of accuracy.

The example also demonstrates that either of the two different combination approaches – equivalent finite elements or the equivalent boundary elements approach – can be equally well applied to solve the problem.

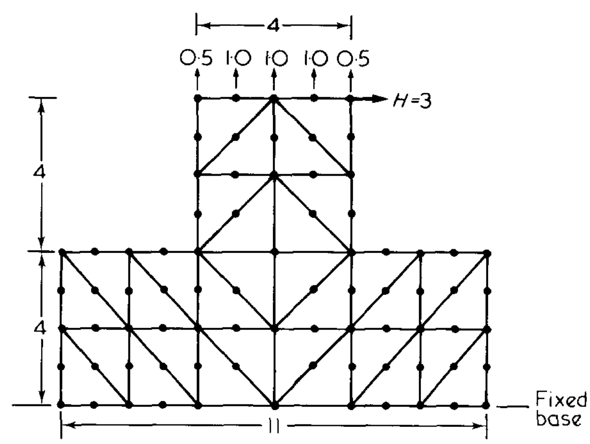

Figure 5 Grid used for finite element method

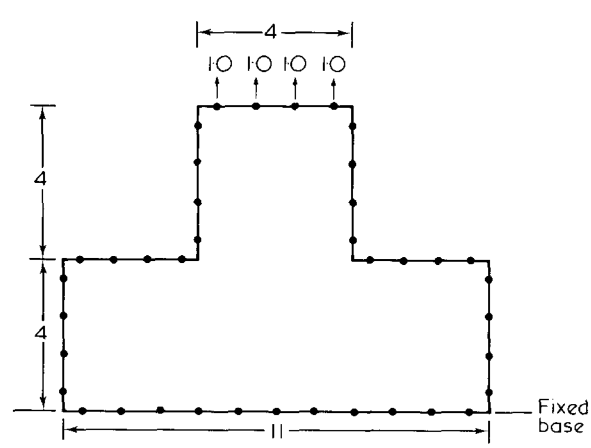

Figure 6 Mesh used for boundary element method

Table 2

	Combination						
	Approach 1		Approach 2		Finite elements only		Boundary elements only
	Results along the top five nodes (*Figures 5, 6* x 10^{-6})						Average vertical displacement
	Displ. *X*	Displ. *Y*	Displ. *X*	Displ. *Y*	Displ. *X*	Displ. *Y*	
Case 1							
	−3.59	33.50	−3.59	33.50	−3.50	33.40	
	−1.35	30.91	−1.35	30.91	−1.35	30.84	
	0.00	32.66	0.00	32.66	0.00	32.62	33.00
	1.35	30.91	1.35	30.91	1.35	30.84	
	3.59	33.50	3.59	33.50	3.59	33.40	
Case 2							
	211.56	−58.66	211.55	−58.67	213.19	−58.58	
	183.06	−0.83	183.05	−0.83	184.67	−0.79	
	165.86	36.36	165.85	36.36	167.43	36.27	—
	161.48	62.32	161.48	62.33	163.17	62.12	
	162.33	97.40	162.32	97.41	164.06	97.20	

(Note: The concentrated load, *H*, cannot be represented accurately in the boundary element method especially with constant elements.)

Example 2

In this example the use of boundary elements to represent a semi-infinite space is examined. This type of problem is of special relevance for soil-structure interaction.

Region 2 of *Figure 7* represents the semi-infinite formulation and was given a semi-circular shape of very large diameter in relation to the loaded segment. Boundary conditions to restrain rigid body movements were applied. The region was also analysed using a finite element grid shown in *Figure 8*, and the results compared to the analytical solution for a semi-infinite plate.[12] They are compared in *Table 3*.

The above results are in good agreement and the differences between the numerical and the analytical solutions can be attributed to the different boundary conditions. The numerical solutions are effectively stiffer than the analytical one because in the former a series of boundary points below the surface are fixed.

Regions 1 and 2 shown in *Figure 9* are then combined, where region 2 can be a boundary element or a finite element domain. The combination of the boundary element region 2 with the finite element region 1 was carried out using approach 1, i.e. by finding an equivalent symmetric stiffness matrix for the boundary element domain. Two loading cases were applied (*Figure 9*): Case 1 considers five concentrated vertical loads along the top and Case 2 considers an additional horizontal load acting at a corner.

The results obtained are shown in *Table 4*.

The combination solution is in excellent agreement with the results obtained using finite elements for the whole domain (*Figure 10*), but it is interesting to note that the foundation was adequately represented using only 37 boundary element nodes as opposed to 163 for the finite element case.

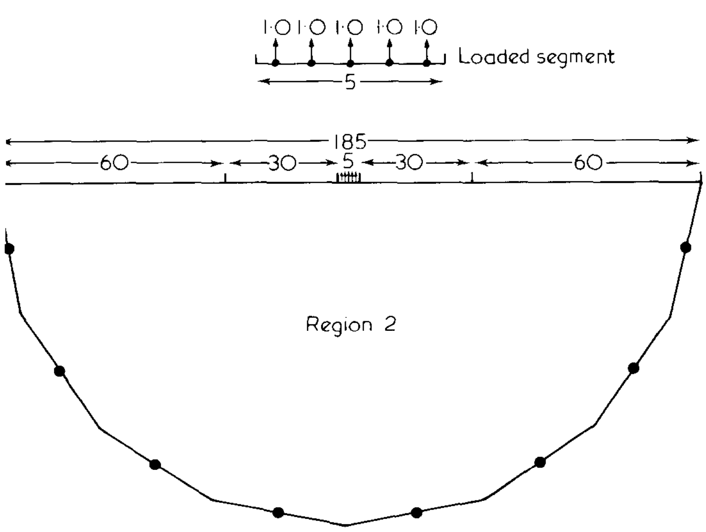

Figure 7 Boundary element mesh

Table 3 Results for semi-infinite foundation under a uniform strip load. $E = 2 \times 10^5$; $\nu = 0.2$

Central displacement x 10^{-6}		
F.E.	B.E.	Analytical
64.8	66.2	71.5

Conclusions

The present paper examines the possibility of combining finite element and boundary element techniques for elastostatics using two alternative procedures. The two approaches are equivalent as is confirmed by the fact that only slight differences, attributed to rounding errors, occur in the results.

Table 4 Vertical displacements along nodes

Vert. displacements along loaded top x10^{-6}			
Load case 1		Load case 2	
F.E. solution	Combination solution	F.E. solution	Combination solution
141	140	−339	−355
134	133	−97	−105
−132	132	135	135
134	133	361	370
141	140	600	617

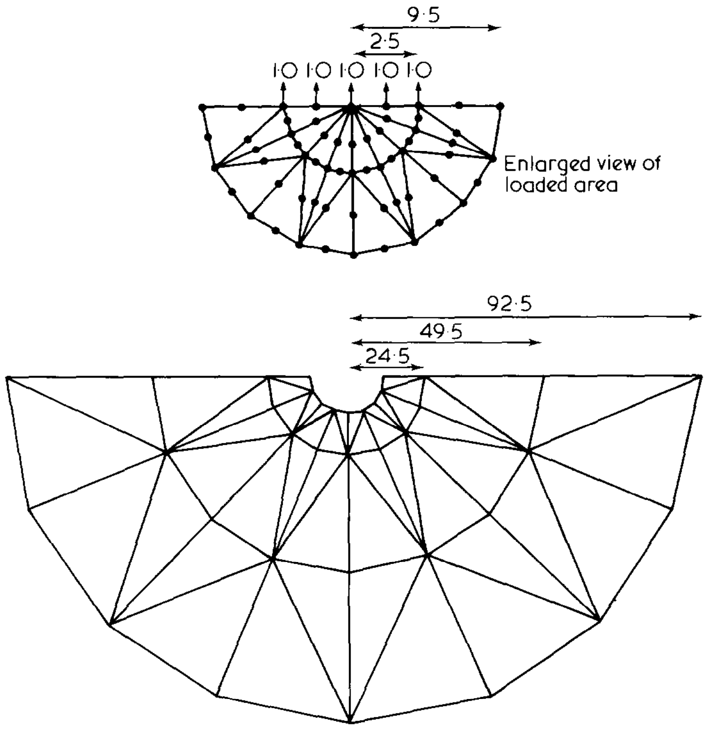

Figure 8 Finite element mesh

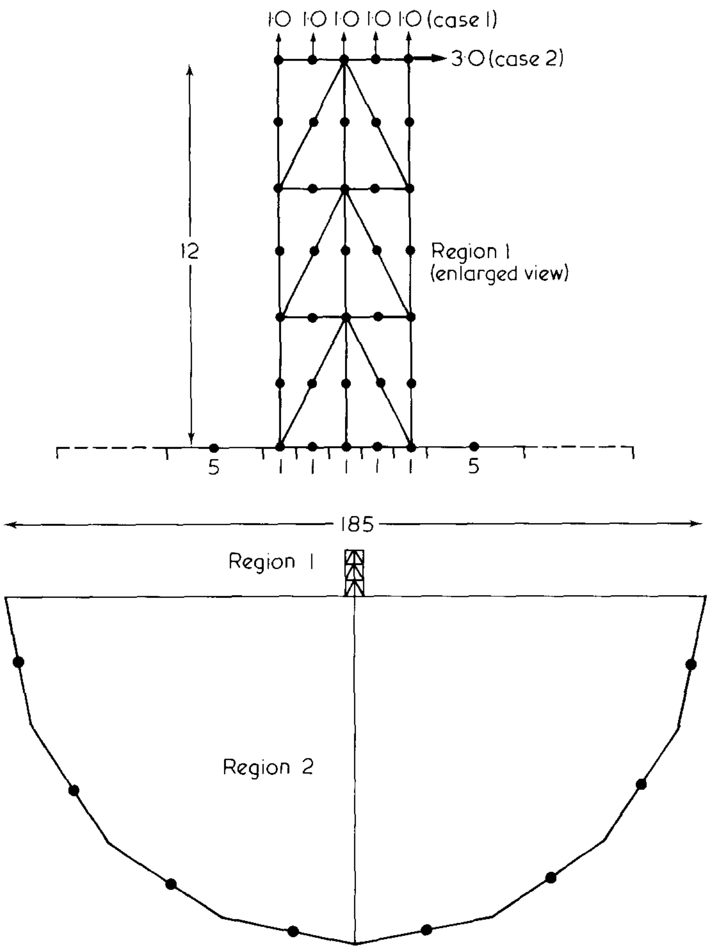

Figure 9 Finite and boundary element combination mesh

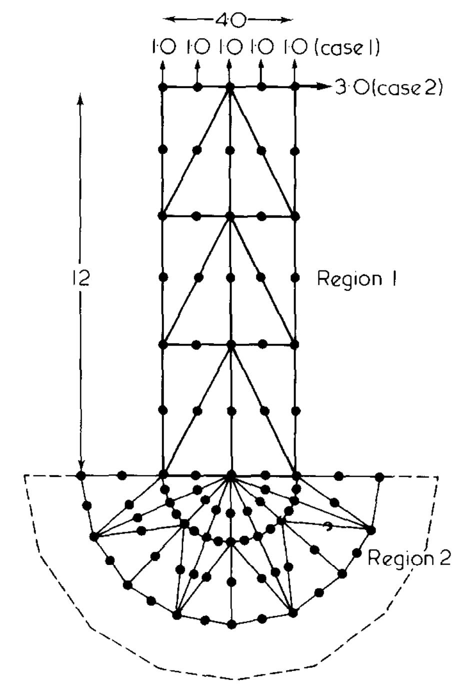

Figure 10 Finite element regions problem. Mesh for two region problem. Rest of grid as in *Figure 8*

The assumptions used in approximating u and p around the boundary for the formulation of the M matrix (equation 37) are proved to be correct due to the agreement of the results obtained when solving the same problem as a standard boundary element problem and as an effective stiffness problem (equations 23 and 44). The validity of the least square symmetrization technique is also demonstrated.

The combination of boundary and finite elements can be carried out using two alternative approaches. The first of the two procedures is in essence a stiffness method and can easily be incorporated into existing finite element packages, although it requires the inversion of the non-banded G matrix. In contrast, the second of the alternatives does not require this inversion, but the system of equations remains asymmetric, and along the interface both the displacements and tractions remain unknown. The implementation of this technique on a large scale commercial level would require development of new computer packages as it cannot be incorporated into existing finite element computer codes.

The ability to represent large domains accurately with relatively few nodes makes the inclusion of boundary elements indispensable for realistic modelling of many engineering problems. Finite element results for regions extending to infinity can be very inaccurate, but these regions can often be well represented using boundary elements. In addition it is possible in many cases to use the fundamental solution for a semi-infinite space, and in this case only nodes along the interface are needed (e.g. at a soil-structure interface). For these problems the inversion of the G matrix is relatively simple and the first approach may be preferable.

Although the results presented in this paper are all two-dimensional, the advantages discussed above are more marked in three dimensions, where the use of finite elements can entail very large systems of equations.

References

1 Rizzo, F. J. *J. Appl. Math.* 1967, **25** (83), 00
2 Cruse, T. A. and Rizzo, F. J. Part I. *J. Math. Appl.* 1968, **22**, 244; Part II. *J. Math. Appl.* 1968, **22**, 341
3 Cruse, T. A. In 'Variational Methods in Engineering' (Eds. C. Brebbia and H. Tottenham), Southampton University Press, 1973
4 Tomlin, G. R. *PhD Thesis*, Univ. Southampton (1973)

5 Lachat, J. C. *PhD Thesis*, Univ. Southampton (1975)
6 Brebbia, C. A. and Dominguez, J. *Appl. Math. Modelling*, 1977, **2**, 372
7 Brebbia, C. A. *Appl. Math. Modelling*, 1978, **2**, 160
8 Brebbia, C. A. and Butterfield, R. *Appl. Math. Modelling*, 1977, **3**, 132
9 McDonal, B. H. and Wexley, A. *IEEE Trans. on Microwave Theory Techniques, MTT-20, No. 12*, December 1972
10 Osias, J. R. *et al.* In *Proc. First Symp. Numer. Analysis Appl. Eng. Sci.* Versailles, France, 23–27 May 1977
11 Mei, C. *Proc. X Symposium Naval Hydrodyn.* 1975
12 Timoshenko, S. P. 'Theory of Elasticity', McGraw-Hill & Co.
13 Zienkiewicz, O. C. *et al. Int. J. Numer. Meth. Eng.*, 1977, **11**, 355
14 Chaudonneret, M. In 'Recent advances in boundary element methods' (Ed. C. A. Brebbia), Pentech Press, London, 1978